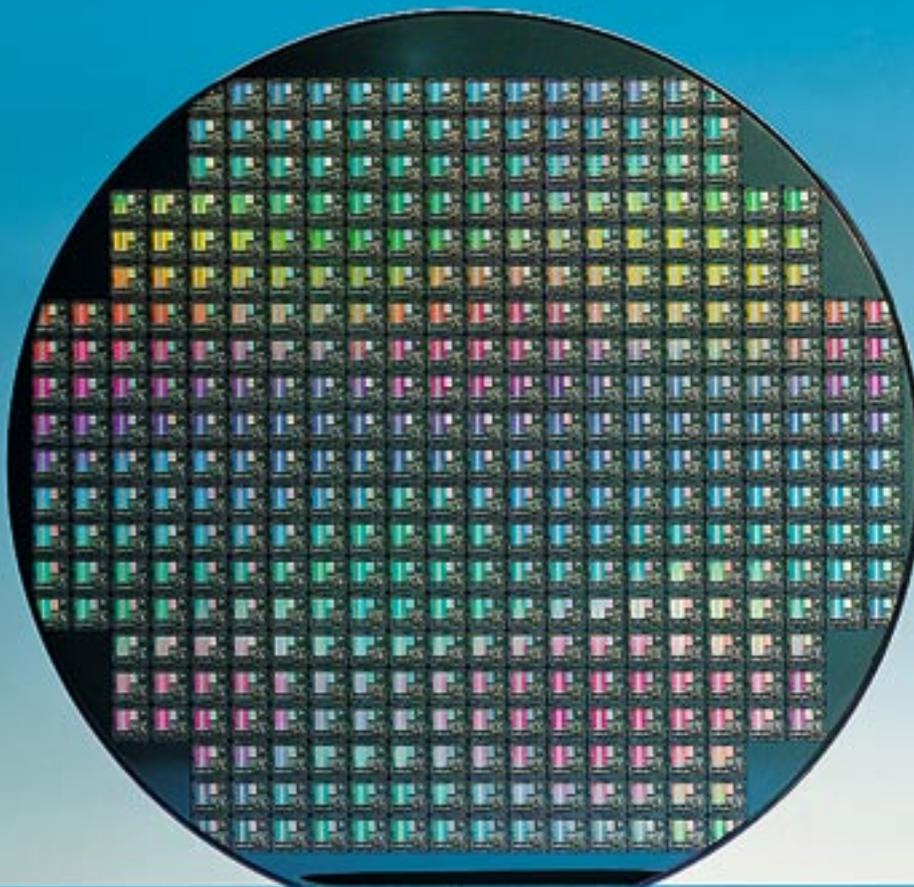




Integrated Circuit/EM Simulation and Design Technologies for Advanced Radio Systems-on-chip



The project

ICESTARS (Integrated Circuit/EM Simulation and Design Technologies for Advanced Radio Systems-on-chip) has received research funding of the European Commission under the FP7 Information and Communication Technologies Programme (ICT), grant agreement number ICT 214911.

Maximum community contribution to the project:

2.8 Mio Euro

Project start:

1 February 2008

Duration: 30 months

Partners involved: 9

Type of project:

Collaborative Project

Project website:

<http://www.icestars.eu>

The ICESTARS consortium comprises two industrial partners (NXP and Infineon), two SMEs (Magwel and AWR-APLAC) and five universities (Upper Austria, Cologne, Oulu, Wuppertal, Aalto).

The project is coordinated by NXP Semiconductors, The Netherlands.



About ICESTARS

ICESTARS addresses a series of critical issues or “bottlenecks” in the currently available infrastructure for the design and simulation of new and highly-complex Radio Frequency (RF) front ends operating beyond 10 and up to 100 GHz. Future RF designs demand an increasing blend of analog and digital functionalities.

The ICESTARS research will accelerate the chip development process in the super and extremely high frequency (SHF and EHF) range to accomplish future demands for higher capacity channels. With today’s frequency bands of approximately 1 to 3 GHz it is impossible to realise extremely high data transfer rates. The key to enable the realisation of single-chip integration of high-GHz wireless modules is resolving the shortcomings in available design flows.

The ICESTARS research area comprises the three domains of RF design:

- time-domain techniques
- frequency-domain techniques
- EM analysis and coupled EM circuit analysis.

The objective

The objective of the ICESTARS project is to enable the realisation of **single-chip integration of high-GHz wireless modules** by developing **integrated simulation algorithms** and **prototype tools** that overcome the barriers in existing design flows.

The project aims at an efficient connection between the frequency domain, where the RF part of wireless transceiver systems is usually designed, with the time domain, where the digital signal processing and control logic are developed. The interactions between devices in the RF front-end at very high frequencies mark the electromagnetics (EM) research area.

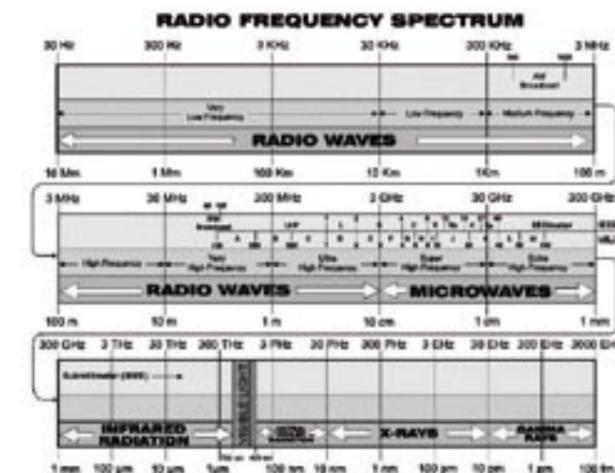
ICESTARS introduces not only novel Computer Aided Design (CAD) and new Electronic Design Automation (EDA) tools, but also the mathematical methods to deal with analog/digital mixed-signal simulation and future challenges in system design and methodologies, thus enabling European developers of RF integrated circuits (RFIC) and semiconductor manufacturers to maintain their top position at the global telecommunication market.

A foray into the SHF and UHF spectrum

Protruding into RF design in frequencies beyond 3 GHz necessitates an ever increasing miniaturisation of components. Today’s SHF and EHF band RF designs are functionally not adequate as accurate simulations of such systems are either inefficient or not available. That is where the ICESTARS research focus is situated.

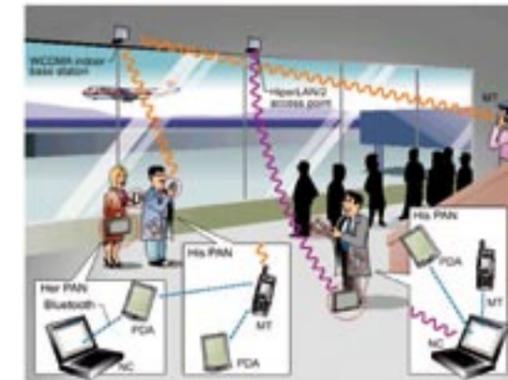
Band	Frequency	Wavelength	Present use
SHF	3–30 GHz	100 mm–10 mm	microwave devices, mobile devices (W-CDMA), WLAN, most modern radars
EHF	30–300 GHz	110 mm–1 mm	radio astronomy, high speed microwave radio relay

Today virtually every RF design consists of multiple complex RF modules combined with ultrahigh-speed digital and analog circuitry in tight proximity on the same circuit board. But the RF design methodologies have not kept pace with advances in RF technology, resulting in delays in the deployment of advanced RF capabilities caused by protracted design cycles.



A new generation of mathematical models

Only a new generation of CAD and EDA tools will ensure the realisation of complex nanoscale designs. It necessitates both new modelling approaches and new mathematical solution procedures for differential equations with largely differing time scales, analysis of coupled systems of time-dependant DAEs and PDEs plus numerical simulations with mixed analog and digital signals. In ICESTARS new techniques and mathematical models working in highly integrated environments are developed to resolve this dilemma.



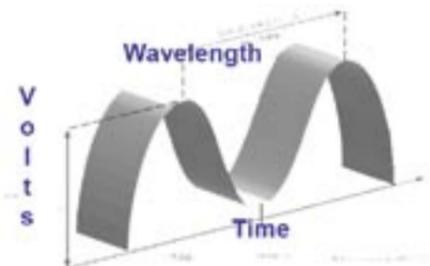
RF design is everywhere

Recent years have seen a phenomenal growth in the field of mobile devices with products combining analog and digital parts to enhance transceiver functionality for radio, GSM emitting for phones that, e. g., offer additional functions such as sensors and cameras. Traditional Radio Frequency (RF) design approaches no longer meet the challenges of next-generation communication products.

Algorithms are everywhere

A lot of everyday technology is based on complex mathematics with algorithms playing a vital role. The best wireless devices are made by the companies with the best mathematicians. And since computers and mobile technologies are increasingly dominant in our lives, algorithms are becoming increasingly important.

In principle, algorithms are just a sequence of steps that examines what is happening and comes to a conclusion – like, e.g., a flow-chart. Every time one downloads music or pictures to one’s mobile or goes online, algorithms lead the way.



Properties of a radio wave

Wireless communication makes use of electromagnetic waves to send signals across long distances. Electromagnetic waves are defined by their characteristics. The amplitude, or strength, can be visualised as its height – the distance between its peak and its lowest point – and is measured in volts. The frequency refers to the number of waves generated in a set period of time and is measured in hertz (Hz).

1 Hz means one wave per second, 1 kHz (kilohertz) means one thousand waves per second, 1 MHz (megahertz) means one million waves per second, 1 GHz (gigahertz) means one billion waves per second and so on.

Modulation

A radio wave conveys no information, but is simply a rhythmic stream of continuous waves. Only when modulated, the RF signals are used as carriers of the information-bearing signals in wireless applications. Modulation is the actual process of encoding information in a radio signal by varying the characteristics (amplitude, frequency or phase) of the radio wave. Phase and amplitude modulation are used to encode digital information (consisting of 0s and 1s) into radio signals.

The RF design flow

Broadband wireless access and mobile devices are just some of the newly-developing services which will need spectrum in the years immediately ahead. Modern RF designs operate at GHz frequencies to maximise link reliability and minimise interference with other services.

Circuits performing at these frequencies are extremely sensitive to, amongst others, active and passive device models, distributed layout parasitics, substrate coupling effects, and supply noise. To enable first-pass success it has become more important than ever to accurately characterise layout and other parasitic effects in RFIC design.

Mixed-signal environment

The structural and functional components of many RFICs are created in a mixed-signal environment and then integrated on chip. Verification of these blocks is performed using circuit simulators to critically assess accuracy. The addition of high-performance circuit simulation technology, combined with a high-frequency simulation component and layout electromagnetic extraction, into the design flow enables first-pass silicon success.

Type of circuit	ICESTARS research area
Oscillator (Colpitts, Pierce, VHF)	wavelet analysis, HB analysis, sweep following, initialisation
Amplifier (LNA, PA)	wavelet analysis, HB analysis, Volterra on Harmonic Balance (VoHB)
Mixer	wavelet analysis, HB analysis, multirate-wavelet and envelope analysis
IQ modulator and demodulator	HB analysis, envelope analysis
Phase locked loop (PLL)	HB analysis, envelope analysis
RF receiver/transmitter	multitone HB analysis
Octo-shape inductor	transient EM simulation
Inductor-oscillator coupling	transient coupling of circuit solver with EM solver

Overview of ICESTARS research application areas

Circuit design and simulation

Circuit simulation is performed in the time and frequency domains to characterise critical performance metrics. Most circuit designers choose a domain depending on the type of circuit and on the



Mathematics for the RF designer's tool chest

The mathematical part in IC design (ICs being miniaturised and complex electronic circuits) refers to the physical mapping of the devices – equations are used to describe the various relations, e. g. positioning, geometry and wiring of electronic components, nonlinear behaviour (transistors), responses in time and frequency domain, inductive coupling and tuning of oscillators.

The overall designs are mathematically complex and call for a lot of electromagnetic coupling. Mathematical equations such as ordinary differential equations (ODEs), differential-algebraic equations (DAEs) and partial differential equations (PDEs) are core to simulate and predict the behaviour of the designed integrated circuits, before the expensive manufacturing process starts.

specifications they need to meet, etc. The simulation is carried out with an optimised transient simulation engine or performing frequency-domain simulation using, e. g., a harmonic balance (HB) engine.

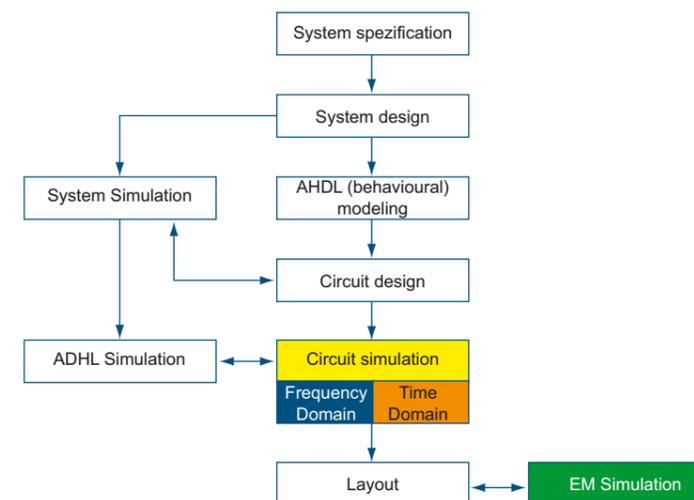
Circuit layout

The next step is to perform circuit layout using either an automated design-rule-driven or connectivity-driven layout. After layout completion electromagnetic simulation is used to provide highly accurate models for interaction of passive components and interconnect. These EM simulation models allow the designer control over the passive modelling process as they can be mixed and matched with the existing models.

Then package parasitics are extracted and their effects added to the circuit simulations. At RF frequencies even the smallest amount of lead inductance can have a significant effect on circuit performance.

Behavioural test benches

The final step prior to tapeout is the full-chip verification in a system (behavioural) test bench. The test bench serves as a framework for more complex simulation. It can be used to simulate the entire RFIC and verify the performance against key wireless specifications. These initial simulations provide information about the expected ideal performance characteristics of the design. They also offer details of the design partitioning and requirements of the individual functional blocks.



RFIC design flow (detail) and mapping ICESTARS research topics



Oscillator circuits

From a mathematical point of view, simulations of oscillator circuits are most demanding and challenging. Modified nodal analysis (MNA) for these circuits leads to typical differential algebraic equations (DAEs) instead of ordinary differential equations (ODEs). This restricts the choice of numerical integration methods which can be used to solve the systems numerically.

A further difficulty is that oscillator systems show nonlinear behaviour causing the signals at different frequencies to influence each other. This causes problems, when coupling different time scales. The oscillators may also exhibit long transients before entering their periodic state. Though these transients are not in the focus of the project, a direct time integration method cannot avoid computing them.

The ICESTARS simulation results are presented for free running oscillators like a Colpitts oscillator, voltage controlled oscillator (VCO) and Pierce quartz crystal oscillator.

New time-domain techniques

An unprecedented transformation in the design and deployment of short-range wireless devices and services is in progress today. Time-domain simulation is used to verify whether a circuit satisfies a given set of performance criteria. New time-domain techniques are needed to make the simulation of the digital-rich circuits more efficient. With higher frequencies, however, “simple” time-domain simulations and existing simulation algorithms are not adequate when it comes to mutual simulation of digital and RF parts.

Traditional RF algorithms were developed at a time when the analog and digital parts were not integrated on the same die. Present RF circuits feature a sharp increase in digital content with analog parts being increasingly replaced by their digital counterparts. The sharp drop in performance of existing RF simulators (especially speed, memory requirements and robustness) necessitates the development of novel algorithms capable to efficiently simulate RF circuits comprising signals with steep slopes as well as multirate (multitone) circuits.

Versatile circuit algorithms

The ICESTARS research of time-domain techniques covers the development of novel algorithms for transient analysis based on wavelet approximations as well as multiscale algorithms. The wavelet approximations are intended to complement and/or replace standard integration formulas such as the well-known BDF techniques.

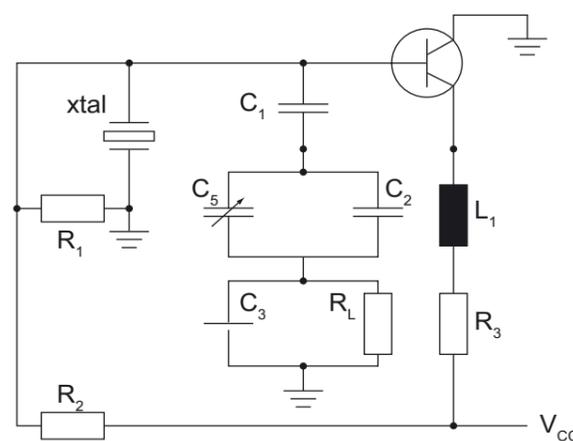


Diagram of a Colpitts oscillator

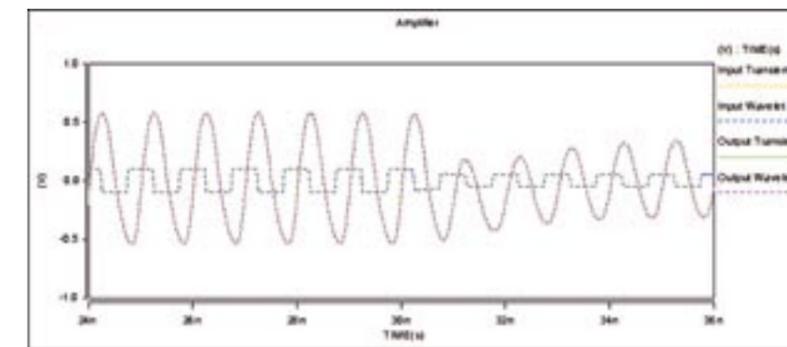
In ICESTARS the validation of these methods is performed for different classes of circuits. The test benches range from LNAs to mixers and oscillators. These versatile circuit examples enable a fair comparison with existing techniques.

Simulation: Novel wavelet-based technique

A new adaptive wavelet-based circuit simulation algorithm has been developed within ICESTARS. Subsequently a prototype of a wavelet solver for circuit simulation has been implemented in the framework of the Infineon in-house circuit simulator.

To realise the new wavelet-based circuit simulation algorithm, the unknown voltages and currents from MNA description are expanded into a wavelet representation, which is determined as solution of nonlinear equations derived from the circuit equations by a Galerkin discretisation.

The wavelet representation is adaptively refined during the Newton iteration until the prescribed error tolerances are met. The resulting approximation requires an almost minimal number of degrees of freedom, and in addition the adaptive approach enables a very efficient numerical computations.



Comparison of the wavelet vs. transient analysis results on the input/output signals of the amplifier

A new generation of mathematical models

For the validation of the new wavelet algorithm the set of typical RF circuits representatives, namely the oscillator, mixer and the amplifier, was used. Accuracy and performance of the wavelet solver have been successfully tested against highly optimised transient analysis algorithms.

The results of the simulations indicate that the wavelet based method may achieve the performance of the standard transient analysis. Since the relatively new wavelet approach has a large potential for optimisation, the ICESTARS partners are optimistic that wavelet analysis will be a valuable tool for circuit simulation in the future.

An entire computer on a single chip

An integrated circuit (IC) is a very small electronic circuit consisting mainly of semiconductor devices, sometimes also called microcircuit or chip. The term “chip” derives from looking like a minute small thin piece of aluminium. Technically, an IC is a set of micro-miniaturised, electronic circuits fabricated on a single piece of semiconducting material. Digital chips are used as processors, memory and for other information processing functions in computers. Chemically, a semiconducting material (silicon in the vast majority of chips) is combined with other materials to alter its electrical properties.

The more functions contained within the chip, the more systems can be miniaturised for handheld use with an ancillary reduction in power. A chip for a telecom application might, e.g., contain a microprocessor, digital signal processor, RAM and ROM and even a graphics processor.

Envelope Simulation

In circuit-envelope simulation, input waveforms are represented as RF carriers with modulation envelopes. Those modulation envelopes are described in the time domain. The input waveform consists of a carrier term and a time-varying term that describe the modulation applied to the carrier. Amplitude, phase, and frequency modulation or a combination of these can be applied.



By using different independent variables for slow and fast time scales the original system has been transformed to a hyperbolic multirate differential algebraic system that allows to reconstruct the solution of the original system. After discretising the slow time scale, a periodic boundary value problem in the fast time scale is solved for each slow time step, the solution is done by wavelet collocation using piecewise linear *ansatz*-functions. To adapt the grid, the solution is represented in terms of a hierarchical set of biorthogonal wavelets that detect the areas with rapidly changing solutions and coarse or refine the grid.

Multirate-wavelet and envelope techniques

In ICESTARS multirate envelope techniques in the time-domain have been developed that exploit multirate RF signals comprising a fast carrier and a slower baseband signal (i.e. envelope). Signal distortions and the bit-error-rate (BER) as well as switching transients are of interest to an RF designer.

Among the typical, practical applications of circuit-envelope simulation is the analysis of amplifier and mixer circuits that operate in pulsed or digitally modulated RF environments.

The simulation problem: On the one hand, the simulation interval must be in the order of the largest relevant time constant or vice versa reciprocally proportional to the lowest frequency of interest. On the other hand, the largest time step is restricted by the highest frequency of interest. According to Nyquist's theorem the sample rate of the waveforms must be at least twice as high as the highest relevant frequency.

Therefore in ICESTARS envelope simulation has been carried further by developing multirate methods based on wavelets dealing with widely separated time scales. Typical applications are circuits, where a periodic carrier signal or where frequency modulation applies.

The ICESTARS envelope simulation has been specifically designed to work even in the case of steep gradients due to digital-like signal structures. An optimal time splitting allows for efficient envelope simulation as well as frequency modulation.

The envelope method, based on embedding the system of ordinary DAEs into partial DAEs, is the only technique with a sound mathematical basis. It exhibits a general mathematical framework that can be adapted to different classes of RF circuits.

In ICESTARS it has been implemented in the Infineon in-house circuit simulator and in LinzFrame and successfully verified for a large class of circuits, namely oscillators and mixers.

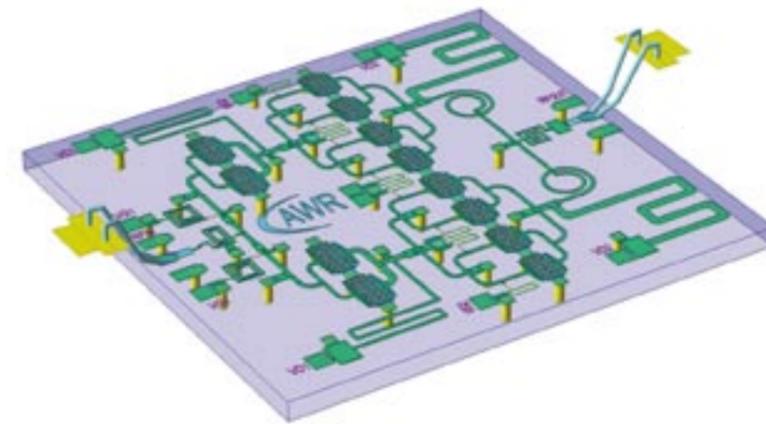
Bridging the gap: frequency-domain techniques

System architects typically develop algorithms in the time domain, which is conducive to creating signal flow algorithms. In contrast, RF engineers deal with the frequency response of the amplifiers, filters, mixers etc. in terms of network parameters as well as frequency-dependent noise and non-linearity.

Frequency-domain simulation is usually carried out using engines such as, e. g., harmonic balance or by mixed time-frequency analysis using an envelope simulator. After a suitable circuit topology is selected, a preliminary transistor-level description of the circuit can be created and simulated. Through an iterative process the design will be refined until the RF block specification has been satisfied.

Objectives of ICESTARS frequency-domain techniques:

- scalable algorithms for RF simulations
- adaptivity in Harmonic Balance
- initial conditions for Harmonic Balance
- improvement of the Volterra-on-HB technique
- development of analysis and dimensioning tools for RF power amplifier (PA).



Layout of an AWR Design Environment from 1W X-band HPA MMIC design

Scalable RF simulation

Within ICESTARS existing frequency-domain algorithms are improved respectively new ones are developed – the development of which is based on scalable algorithms for RF design. Scalability in general is applied to enable RF simulations for large circuits.

The algorithms used in Harmonic Balance and its noise analysis are optimised for different types of circuits and available computational resources using the most accurate initial frequency estimates. Volterra-type analysis is used for detailed distortion analy-



Frequency modulation

Mathematical modeling of circuits is based on implicit systems of ODEs or on systems of DAEs. These systems are not efficient when it comes to modulating RF signals with widely separated time scales. To efficiently represent such RF signals a multi-dimensional model is needed: a system of multirate partial differential algebraic equations (MPDAEs).

For the efficiency of such frequency-modulated signals it is crucial to define an adequate local frequency function. Here the concept of MPDAEs enables an alternative strategy for simulating electric circuits in RF applications. Techniques based on minimisation are feasible for the determination of the free modelling parameters in case of frequency modulated signals.

In ICESTARS a Colpitts oscillator has been simulated using an initial-boundary value problem of the multirate system.



Harmonic Balance (HB)

Harmonic Balance obtained its name from a simulation technique where the computer iteratively adjusts the magnitude and phase of all harmonic signals flowing around in a circuit until the sum turns to zero at every node to obey Kirchhoff's current law. When this condition is achieved, "harmonic balance" is achieved and the nonlinear circuit is solved.

The harmonic signals are the complex Fourier coefficients of signals in the circuit being represented in the frequency domain. A square wave signal of, e. g., 1 kHz will have many higher harmonics that are multiples of the fundamental 1 kHz tone. When these harmonics hit a nonlinear device such as a transistor, more harmonics of the fundamental tone are generated. The more nonlinear a circuit is, the more harmonics its solution contains and therefore it requires more memory and time to achieve HB in the solution process.

Complexity further increases, when multiple fundamental tones are injected, and frequency mixing occurs to generate even more harmonics for the simulator to achieve HB convergence. This is where advanced mathematical algorithms are developed.

Design and simulation

It has proven a time-consuming business to achieve a reasonable estimate for the initial conditions for distortion analysis of free-running oscillators. The purpose of diagnostic tool development is to provide the user (and the algorithm itself) with information about the HB analysis state that has not been available previously. The development has taken place in both the APLAC and the Infineon in-house circuit simulator. The APLAC simulator was improved to meet any kind of adaptivity.

Within ICESTARS new algorithms have been developed, intended to find the operating frequency of a free-running oscillator, thus improving both speed and robustness of harmonic balance analysis. One algorithm, using available pole-zero analysis methods to determine the dominant poles of the system, calculates a good initial estimate for the operating frequency. A different approach tries to find an initial frequency estimate from optimisation techniques.

Improved reliability and robustness

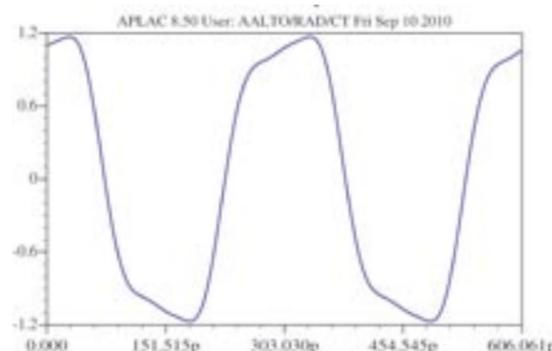
The new developed HB analysis of an oscillator

- adds an artificial excitation, i.e., probe element to the circuit
- uses initial transient analysis to obtain voltage waveform of the probe. Two methods named FFT and ZeroC for frequency and amplitude estimate frequency and amplitude. An extrapolated version of ZeroC is in use in Pstar of NXP.
- A more robust nonlinear solver was developed at NXP based on modern *eigenvalue* techniques.

Initial values: fosc=3.0 GHz, vosc=1.0V		
method	HBITER	CPU(s)
old	2259	110.6
FFT	47	6.2
ZeroC	31	1.4
Estimated fosc and vosc		
FFT	2.7 GHz	1.9 V
Zero C	3.1 GHz	1.17 V

VCO oscillator: output waveform

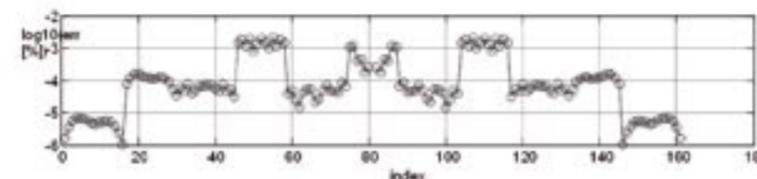
The usage of these new methods improves reliability and robustness of HB oscillator analysis as the user does not need to supply accurate initial values.



Design of linear power amplifiers

Volterra-type analysis is used for detailed distortion analysis especially needed to design linear power amplifiers. It has been extended to multi-device circuits with significant speed-ups.

The main advancement in the VoHB technique has been the APLAC AIF interface enabling important new features, amongst others, improved convolution algorithm (applicable with multitone excitations), direct calculation of the response of the circuit for the calculated nonlinear Volterra current or the calculation of distortion contributions for each HB frequency.



2-sided spectra of the convolution, accuracy of the fitting against HB.

The VoHB algorithm is implemented using a function interface AIF that allows C-language access to many internal functions and data structures of APLAC, including the complete circuit matrix, and input and output signals of the VCCS sources that constitute all device models in APLAC.

The cause of distortion

The analysis results of VoHB are used to find the dominant cause of distortion, available cancellation mechanisms, design guidelines to minimise distortion, e.g., by tuning the harmonic matching impedances and causes of bandwidth-dependent memory effects, that complicate the design of digital pre-distortion systems, for example.

VoHB analysis tests have been carried out for power amplifier circuits. First, tests have been run using the old APLAC input language based models. These results were then compared to the results that the new APLAC analyzer interface (AIF) model produces.

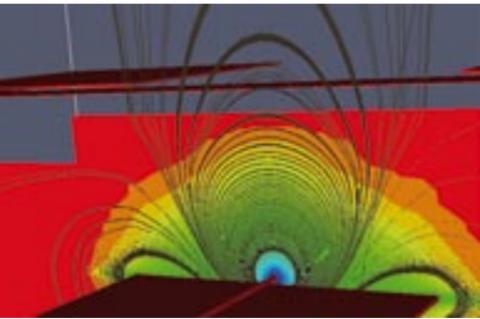
For the first time, a truly generic multi-device VoHB algorithm was coded and has been tested for circuits larger than plain single-transistor power amplifiers.



VoHB

Volterra-on-Harmonic Balance (VoHB) is a technique developed by J. Aikio at Oulu university. It enables to see a detailed construction, dominant causes and cancellation mechanisms of nonlinear distortion in analog circuits. VoHB is further evolved in ICESTARS.

The distortion analysis algorithm called Volterra on Harmonic Balance (VoHB) breaks the total nonlinear distortion (given by harmonic balance) to a vector sum of smaller components so that the dominant causes, mixing and cancelling mechanisms can easily be seen. This is done for each nonlinear element and a polynomial presentation is fitted in the frequency domain. These polynomial expansions are then used in Volterra analysis to calculate different mixing mechanisms from one tone to another. For both fitting and the expansion a new efficient frequency domain convolution algorithm is used.



EM Simulation

EM simulation is needed to model high-frequency and high-speed circuits and devices and absolutely necessary to accurately verify the interactions between devices in the RF front-end at very high frequencies. Regarding the mere scale of EHF band communication front-ends, it is impossible to run full simulations at the level of detail an EM analysis offers. Therefore EM analysis only looks at a small part of the total design, while the rest of the circuit is simulated at a higher level of abstraction.

To this end, the state-of-the-art RF circuit simulation has to be adapted in order to deal with the specific demands to exactly capture the interaction between neighbouring devices. In particular the frequency response calculation requires the ability to inject electromagnetic energy in a device mode.

Leverage EM analysis and simulation

The ever increasing miniaturisation of future circuits realised in the physical modeling necessitates the simulation of electromagnetic circuits – an entirely new mathematical undertaking.

At higher frequencies parasitics effects are difficult to model. EM simulation is commonly used to accurately model such structures. However, when circuits include the use of lumped components such as transistors, diodes, and capacitors, most engineers abandon EM as being too difficult to set up and run large multi-port problems.

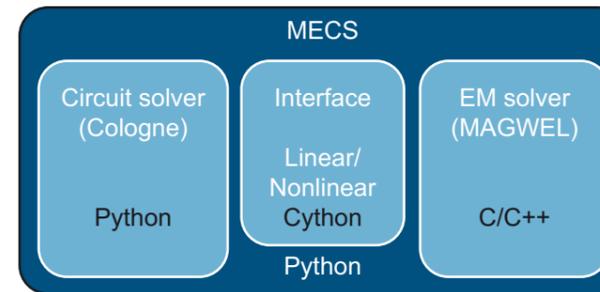
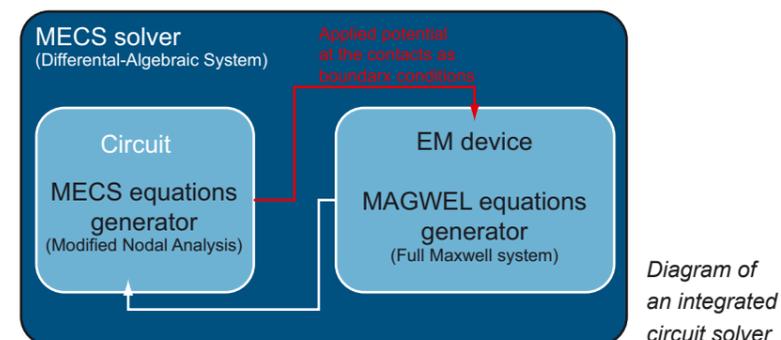
The ICESTARS expertise in circuit-EM simulation aims at providing tools for RF designers enabling them to simulate nonlinear performance of circuits with EM-accurate modeling of distributed components, thereby taking full advantage of the accuracy of EM simulation and the generality of non-linear simulation.

The objectives of ICESTARS EM and coupled circuit research ground are:

- EM analysis
- coupling of circuit simulation and EM simulation
- simulation of selected devices and their coupling using existing tools
- tool development: RF boundary conditions and transient simulation tools
- definition and characterisation of reference benchmark structures.

The ICESTARS mathematicians supply the electromagnetic circuit simulations that are coupled with the components' simulation software of an industrial partner. This is realised by coupling electromagnetic simulation to DAE-solvers for transient simulation.

Coupling of circuit simulation and EM simulation



Coupling of MECS (mos-fets modeled by equivalent circuits) with EM solver via Cython

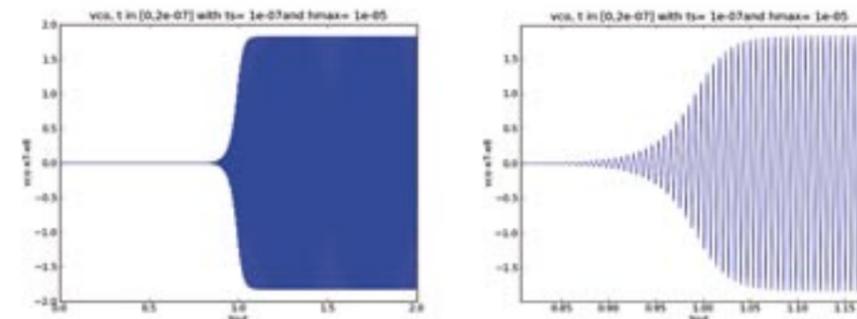


The capability to “experiment at wish” with different mixed mode set-ups was scheduled parallel to main-stream developments in the ICESTARS EM analysis.

The “stand-alone” MECS solver has been transferred from a Matlab environment into a Python framework. The included BDF solver was extended to handle general differential algebraic systems given in a stability-preserving formulation.

Additionally, a general linear method solver has been included into the MECS package.

Magwel provides the equations for the EM devices, while MECS provides the equations describing the circuit topology. It is technically realised by MECS transmitting a coupled system of network equations and equations for the electronic building blocks. The coupled system is perceived as a monolithic item and solved directly.



1.8 GHz VCO Oscillator results (left: full image, right: enlarged detail)

A communication bridge

Cython is used as a communication bridge between C/C++ and Python. The data is exchanged via pointers to vectors and matrices. A shared library generates the EM solver functionality.

The coupled system is solved with a DAE solver (BDF) as a time integrator using adaptive stepsize control and order selection strategies.

The ICESTARS approach models ICs and EM devices using a monolithic approach. This leads to a system of DAEs (describing the circuit and its topology) and PDEs (describing the EM devices) with Maxwell equations describing the EM system.

After spatial discretisation of the Maxwell equations via finite volume scheme, a DAE (with a properly stated leading term) can be obtained resulting in a DAE that now describes the entire system.

Assumptions

The creation of a communication infrastructure that allows the user to communicate anything, any time, anywhere, to anybody, in an affordable, interactive and intuitive way caters in the RF spectrum for an immense range of wireless applications.

Wireless broadband and enhanced mobile phone services are all lining up to be launched.

Technology developments constantly increase the scope for more efficient use of the spectrum by improving encoding and error correction systems.

For digitally controlled or digitally enhanced RF building blocks it is necessary to simulate the digital parts with the RF parts together. However, the existing simulation algorithms are either too slow or too limited to be of practical use for these strongly non-linear transceiver systems.

The technical realisation:

- MECS starts the EM solver (initialisation, memory allocation etc.)
- For every time step MECS calls the EM solver to address the Maxwell system and uses the EM solver's output to calculate integrative equations.
- After time integration MECS stops the EM solver (deallocation of memory, etc.).

The validation requires that the transient mixed-mode equations can be solved using the standard linear solver methods and that the number of iterations does not exceed the required number for solving the time-independent system.

Conventional equations for circuits in general neglect physical effects such as EM fields. They only try to re-build complex building blocks using single parameters – a procedure lacking efficiency as there might be up to 800 parameters. In addition, in the SHF and UHF range models are becoming ever more sensitive.



Inductor loop

The ICESTARS model developed by Magwel and University of Cologne intends to solve these problems by modeling the building blocks using partial differential equations to better project the physical complexity of the models.

Tying it all together

Mobile expansion bands in other parts of the spectrum have been allocated to allow the delivery of additional services by mobile. Already several mobile operators have made TV services available on mobile phones. As technologies scale to smaller technology nodes, the adoption of innovative methods in RFIC design flows is no longer a question of if, but when.

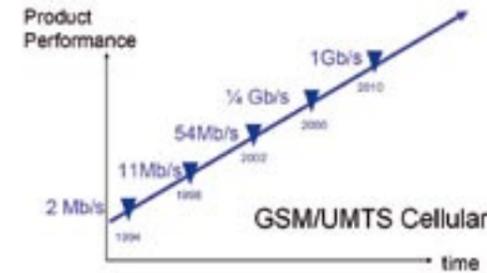
RF circuit-design performance has to be verified against the functional-block specifications. To simulate system-level performance, one or more test benches must be created to generate the RF stimulus and extract measurements. Behavioural test benches serve as the framework for more complex mixed-level simulations allowing to detect and mitigate problems at an early stage of the design process as well as to perform corrective measures.

In ICESTARS the industrial partners Magwel, AWR-APLAC, NXP Semiconductors, Infineon Technologies and the Fachhochschule Oberösterreich (Upper Austria University of Applied Sciences) have as a proof of concept implemented the academically developed mathematical analysis methods in real-life simulation and/or industrial use cases.

A number of relevant test cases from the repositories of the partners, who are end-users, has been set up to compile an inventory. The simulation results of the tools and algorithms developed within the project have been compared against the results obtained with commercial and/or public domain tools already in use. The ICESTARS validation has successfully covered the complete functionality of the tools that have been developed.

ICESTARS publications and presentations

ICESTARS results are prominently made available to both the research community and the broader public on the project website and via publication in leading journals, the organisation of workshops and symposia. The achievements are presented at major conferences such as SCEE or ECMI (an published in series "Mathematics in Industry" by Springer). The conference presentations and publications as well as additional information are available on the ICESTARS website: www.icestars.eu.



Increase in product performance GSM/UMTS cellular

Impact

The scope for new services to be in the future made available in many parts of the RF spectrum is exciting. But that adds to the economical and technological challenges of RF design. Technology design, efficient use of bandwidth, end user demand, availability of receiver equipment, investment in infrastructure and many other technical and market conditions have to be examined to make appropriate judgements.

The ICESTARS project is a combined effort of leading European semiconductor companies and design tool providers together with European universities to harness the power and promise of nanotechnologies towards a networked society.

ICESTARS intends to provide more effective IC and EDA design tools to the European semiconductor industry that will have severe impact on Europe's current strength in this important high-tech area – thus enabling the European market to keep up with global competition.

The ICESTARS consortium

c/o Jan ter Maten
NXP Semiconductors
High Tech Campus 46
5656 AE Eindhoven
The Netherlands



The ICESTARS work packages

Time-Domain Techniques

- Wavelet-based algorithms for mixed analog/digital simulations
- Multirate methods based on wavelet approaches
- Multirate envelope methods

Fachhochschule Oberösterreich

Bergische Universität Wuppertal

*Infineon Technologies/Qimonda**

NXP Semiconductors

Contact:

hans-georg.brachtendorf@hagenberg.at

Frequency-Domain Techniques

- Scalable Algorithms for RF simulations
- Adaptivity in Harmonic Balance
- Initial Conditions for Harmonic Balance
- Improvements of the Volterra-on-HB method
- Developing analysis and dimensioning tool for RF power amplifier

AWR-APLAC

Aalto University School of Science and Technology

University of Oulu

Bergische Universität Wuppertal

*Infineon Technologies/Qimonda**

Contact:

taisto.tinttunen@aplac.com

EM Analysis and Coupled EM-Circuit Analysis

- Simulation of selected devices and their coupling using existing tool conditions and transient simulation tools
- Coupling of circuit simulation and EM simulation
- Definition and characterisation of reference benchmark structures

Magwel N.V.

University of Cologne

NXP Semiconductors

Contact:

wim.schoenmaker@magwel.com

Validation

- Tool development test set and data format specification
- Validation of developments and final tuning

NXP Semiconductors

All ICESTARS partners

Contact:

rick.janssen@nxp.com

The ICESTARS cooperation between mathematicians and engineers from academia and industry did lead to new methods for designing future products with mathematics inside.

Bold = Workpackage leader

* = Infineon has in September 2009 superseded Qimonda (until March 2009)

© 2010 The ICESTARS consortium
Edited and designed by Yvonne Havertz, Cologne
Responsible according to the press law:
Prof. Dr. Caren Tischendorf
c/o University of Cologne, Mathematical Institute
Weyertal 86-90, D – 50931 Cologne